CFD Simulation Study of Natural Gas Pipe Failure

Z.A. MAJID\textsuperscript{a}, R.MOHSIN\textsuperscript{a,*}, M.Z. YUSOF\textsuperscript{b}

\textsuperscript{a} UTM-MPRC Institute for Oil and Gas, Universiti Teknologi Malaysia, 81310 UTM Johor Bahr, Johor Darul Takzim, Malaysia.
\textsuperscript{b} School of Engineering, Universiti Tenaga Nasional, 43000 Bangi Selangor, Malaysia.
*Corresponding author: rahmat@petroleum.utm.my
http://www.yourwebaddress.xx

Abstract: - A failure of API 5L X42 steel pipe was studied using Computational Fluid Dynamics (CFD). By reviewing the existing design and construction data, visual inspection and pipe material analysis, it was predicted that high pressure water jet from leaked water pipe that had mixed with surrounding soil to form water soil slurry with high erosive properties would cause a severe impact on the pipe surface causing the loss of pipe coating materials. This phenomenon eventually caused rapid thinning of the steel pipe body, which later led to its failure. This failure was further studied using CFD simulation. Simulation results were then compared with the actual failed specimen from incident site. CFD simulation results were found able to accurately predict the location, rate and the extent of erosion failures on the pipe surfaces, as compared with actual instances.

Keywords: Pipeline; slurry erosion; pipeline failure; water jet impact: CFD.

1 Introduction

Erosion is defined as an abrasive wear process resulted by a repetitive impact of solid particles entrained in a moving fluid against a surface, causing a removal of material from that surface.\textsuperscript{1} Bitter defined erosion as “material damage caused by the attack of particles entrained in fluid stream system impacting the surface at high-speed”. Solid particle erosion, or slurry erosion, is a term to describe an effect of material removal by mean of particles impact in suspended fluid.\textsuperscript{2} It has been reported to be one source of a problem in many rotating equipment in various prime movers, such as gas turbine, cyclone separators, boilers, and pump. Sand, which is normally used as part of backfilling material, can form water sand slurry in the presence of water. The impact of this slurry has been proven to cause metal loses and metal thinning, in which eventually lead to pipeline failure.\textsuperscript{3,6}

A study by Majid et al.\textsuperscript{3}, Majid et al.\textsuperscript{7}, Majid and Mohsin\textsuperscript{8} showed that a leak of high pressure water pipe in a mixture of soil and sand could create erosive slurry impact on nearby pipes, which may cause serious failure. Detailed investigation conducted in this study proved that high pressure water impact from leaked water pipe could cause local wall thinning to the steel pipe body, and finally produce pin hole leaks.

Simulation of flow condition plays an important role to theoretically determine the effect of flow dissipation in any physical body environment surrounded by liquid base system. It provides an economic option to visualize flow patterns and identify physical parameter variables throughout the contouring zone. Therefore, in order to provide clear explanation of the failure mechanisms, the failure analysis of the pipe was studied using CFD simulation. In this study, the failure incidents as reported by Majid et al., Majid and Mohsin\textsuperscript{8} were used as case studies to explain the erosion failure process of the said pipe.

2 Description of the Failed Specimen

After being impacted, there was a small hole on the surface of the carbon steel pipe, about 10 mm in diameter, located in the middle part of the eroded section. The size of erosion section was around 50 cm, smooth and free from any rust (Fig. 1). The original condition of the exposed part, immediately after the incident, was found to be free from coating materials, and in fact was clean, smooth and shining,
as shown in Fig. 1. This phenomenon was reported as an act of continuous wet sand/soil blasting onto the pipe surface for some period of time, causing surface thinning. The direction of the hole was facing the outer part of the pipe.\textsuperscript{3,8}

![Fig. 1. Failed specimen of API 5L X42 carbon steel pipe](image)

### 3 CFD Simulation

The main objectives of this investigation were to determine the abrasion/erosion effect on the gas pipelines due to the impact of high pressure jet resulting from failure of the adjacent water pipeline. The jet impingement on the pipe test sample was studied in detail and the abrasion rate as indicated by the shear stress distribution in the test sample was investigated. In the water jet simulation, the operating pressure was set to 10 bars at constant mode. No gas jet flow was involved since the primary intention of this part of the simulation was to obtain the flow behavior on how the jet flow affects the steel pipe. In order to simplify the case study, only steady condition was applied. Under the steady condition, it was assumed that computational domain was fully immersed in homogeneous water distribution.

#### 3.1 Mesh Generation

In the present analysis, structured hexagonal multiblock mesh was used in the case of Perspex model. For the test rig model, because the geometry was complicated, multiblock-structured –unstructured hybrid mesh was adopted, whereby unstructured mesh was used in the regions surrounding the test specimen, and structured mesh was used in other regions. The mesh was constructed using the GAMBIT mesh generation routine. High mesh concentration was set around the orifice. Fig. 2 shows the hybrid mesh used for the test rig.

#### 3.2 Boundary conditions

The boundary conditions at wall boundaries, inlet and exit boundaries needed to be set. As explained in the previous section, plane z=0 was set to be symmetrical. Apart from that, the top boundary which represented free surface in real flow was also set to be symmetrical. All other wall boundaries were set as smooth walls. The inlet and outlet boundaries represented the position of the flow coming and leaving the flow domain. The boundary conditions used as inlet and outlet are summarized in Table 1 below.

<table>
<thead>
<tr>
<th>Orifice Diameter (mm)</th>
<th>Inlet Boundary</th>
<th>Outlet Boundary</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inlet velocity (m/s)</td>
<td>Turbulence Intensity (%)</td>
</tr>
<tr>
<td>1.00</td>
<td>0.18</td>
<td>10</td>
</tr>
<tr>
<td>3.00</td>
<td>1.90</td>
<td>10</td>
</tr>
<tr>
<td>5.00</td>
<td>5.50</td>
<td>10</td>
</tr>
<tr>
<td>7.00</td>
<td>10.50</td>
<td>10</td>
</tr>
</tbody>
</table>

![Fig. 2. Hybrid mesh used for the test rig](image)
4 Results and Discussion

4.1 Simulation of water pipe leak
Initially, the failure of the asbestos water pipe was found able to create a small sized hole. The failure started as a small crack, possibly longitudinal. As for that, the simulation was started when the crack size achieved an effective diameter of 1.5 mm. The flow trajectory of the water jet is as shown in Fig. 3a. As the effective diameter increased to 3.0 mm, it was observed that the force of the jet impacting on the steel pipe surface had gotten much more significant (Fig. 3b). The water-slurry was strong enough to cause coating losses on the steel pipe surface. The flow trajectory was also observed to mostly move along the upper surface of the steel pipe. Fig. 4 shows the mesh used and predicted velocity distribution for 3 mm rupture.

![Fig. 3](image1.png)
**Fig. 3.** Predicted velocity distribution for (a) 1.5 mm rupture (a) and (b) 3.0 mm rupture

![Fig. 4](image2.png)
**Fig. 4.** (a) Mesh used and (b) predicted velocity distribution for 3.0 mm rupture

The following Fig. 5 indicates the effect on the velocity contour with the increase of pipe puncture pressure.

![Fig. 5](image3.png)
**Fig. 5.** Velocity distribution at (a) 6 bar and (b) 10 bar

Fig. 6 presents the simulation results in 3-D format. The velocity upon impact at the NPS 8 pipe body was estimated to be around 4 to 7 m/s. This agreed with the fact that together with the water flow, the sand and soil particles could have constituted a considerable erosive effect onto the pipe surface coating and even the pipe body itself. The simulation results also agreed with the observed conditions of the pipe surface conditions after excavation of the site, as shown in Fig. 1.

4.2 Jet flow pattern and erosion behavior on the pipe
In order to study the jet flow pattern and erosion behavior on the pipe, a simulation using two-fluid model was performed. In this modeling approach, sand was modeled as another fluid with density and viscosity twice of that water. The two fluids were not allowed to mix. Initially, the whole domain was set as to consist of only sand, and then the water jet was allowed to enter the domain at higher velocity. Fig. 7 shows the contours of water concentration in the domain after sometime. The blue color denotes regions of high concentration and the orange region denotes a region of low concentration. The region of medium concentration is denoted by green color. The formation of water tunnel could clearly be seen. As expected, higher density sand tended to settle at the bottom.

![Fig. 6](image4.png)
**Fig. 6.** 3-D Water jet simulation of (a) velocity trajectories and (b) pressure contour

![Fig. 7](image5.png)
**Fig. 7.** Concentration distribution of water and sand in the perspex model using two fluid CFD model
The interaction of water and sand caused the sand to lose its compactness, and sands with higher density settled at the bottom due to the influence of the gravitational force, while the sand with higher density settled at the top. The water jet must have created a low pressure region in the upper region and caused the sand to move downward, creating a void region. The actual location of the void region relative to the jet exit plane was determined by the initial stability condition of the sand system.

In order to understand the water jet impingement behavior, a virtual test rig was established. Fig. 8 shows the virtual model of the test rig and the contours of velocity on two planes that passed through the water jet. The formation of the water jet could clearly be seen to impinge directly onto the specimen.

![Virtual model of the test rig and velocity distribution showing the water jet impingement on the pipe test specimen](image)

**Fig. 8.** Virtual model of the test rig and velocity distribution showing the water jet impingement on the pipe test specimen

Fig. 9 shows the development of the water jet and the process that led to the impingement of the water jet on the test specimen. It is interesting to note that, as seen in Fig. 9(b), the velocity at the point of impact was lower than the surrounding velocity. This could clearly be seen in the enlarged view as shown in Fig. 9(c). Also superimposed on the jet was the velocity vector. The behavior had a direct impact on the pipe abrasion behavior, as it was expected that the water jet would entrain sand from the surrounding region and cause a sand blasting effect on the pipe. As the velocity was higher in the area surrounding the point of impact, the abrasion rate would be higher. To investigate this further, the shear strain rate distribution on the pipe was plotted, as shown in Fig. 10.

![Strain rate distribution on the surface around the point of impact (b) as compared to (a) actual failed specimen at incident site](image)

**Fig. 10.** Strain rate distribution on the surface around the point of impact (b) as compared to (a) actual failed specimen at incident site

### 5 Conclusion

Water jet mixes with soil to form water slurry with high erosive properties. This mixture, when impacted upon pipe surfaces, can cause the loss of pipe coating materials. Eventually, corrosion will quickly ensue and material loss will be rapid because of the continuous erosion of oxidized material that happens simultaneously. This phenomenon explains the common rapid thinning of steel pipe body which later leads to its failure. Simulation results from both the CFD support these hypotheses as they match the actual instances.

### Acknowledgement

Authors are most grateful and wish to thank all members of UTM-MPRC Institute for Oil and Gas (formerly known as Gas Technology Centre (GASTEG)) Pipeline Failure Unit and CFD team from Universiti Tenaga Nasional (UNITEN) for their significant efforts and commitments in conducting the experiments and simulation throughout the research period. Gas Malaysia Bhd. (GMB) is also highly acknowledged for providing pipe samples and financial support.

### References: